Using Capture Schematic for PCB Layout

The schematic should have already been created and simulated ensuring that the design works.

<u>In Capture</u>

1. Open the schematic that will be designed in Layout.

2. In the Project Manager view, select the design or schematic (either will work).

3. From the menu bar, select **Tool-> Create Netlist.** The **Create Netlist** box will appear.

4. Select the **Layout** tab. Ensure that your Capture file is selected as the **Netlist File**.

5. Ensure that it has .**MNL** extension and click **O.K**.

<u>In Layout</u>

6. Select **File->New** in the menu bar.

7. Click **File ->New**. The **AutoECO** dialog box pops up.

8. For "Input Layout TCH or TPL or MAX file", browse for **default.tch** (not _defualt.tch) in **Cadence -> PSD_15.0 -> Layout_Plus -> Data**.

9. Browse for your netlist file created in Caputre, for the "Input MNL netlist file" row.

10. For "Output Layout MAX file", a "**filename.max"** file will be automatically created.

11. Click **Apply ECO**. Click **Cancel** on the File B (new netlist) box pops-up

12. Click **Accept this ECO** in the **AutoECO** report window. Click **OK** when process is complete.

13. In some cases with dip-package (op-amps, microcontrollers, etc) footprints, Layout may not find the exact one corresponding to Capture.

14. So the **Link Footprint to Component** window may pop-up.

15. Click on **Link Existing Footprint to Component**. A pop-up window will request a new footprint that can replace the current footprint.

> *For your information:* ***Capacitor****,* ***Resistor*** *and* ***Diode*** *footprints are in the libraries* ***TM_CAP_P****,* ***TM_AXIAL*** *and* ***TM_DIODE*** *respectively. Footprints for* ***Op-Amps****,* ***Opto-Isolators*** *or any other multi-pin chip are in the* ***DIP100T*** *library.* ***Transistor*** *(and* ***Triac****) footprints are in the* ***TO*** *library.*

16. Select the required component footprint and click O.K.

17. Place the footprint within the rectangular outline.

18. Place the design components so that they are evenly spread out. This will make routing and tracking easier later on.

<u>Routing (Connecting pins)</u>

19. Click on the **Connection Tool** in the tool bar.

20. Right-click the mouse and select **Add** from the drop menu.

21. Place the crosshair cursor on a component pin and click the left mouse button.

22. Drag the cursor to the next pin to be connected and so on.

23. Right-click and select **End Command** to end a connection sequence, and start a separate component connection.

24. When all components have been connected, select **View Spreadsheet** in the toolbar. A pop-up menu will appear.

25. Select **Nets**. The Nets spreadsheet will appear.

26. Select the "Width" cell. All the cells in that column should be highlighted.

27. Right click the mouse and select **Properties** from the pop-up menu. The **Edit Net** dialog box will appear.

28. Change **Min Width** to track width 23 (mils). Change **Conn Width** to 50 (mils).

Change **Max Width** to 65 (mils). Click **O.K**.

29. All the widths should change to the entered values in each row. Close the spreadsheet.

30. Next ensure the pad widths are 80mils.

31. Select **View Spreadsheet** in the toolbar. A pop-up menu will appear.

32. Select **Padstacks**. The Padstacks spreadsheet will appear.

33. Select the **Pad Width** and **Pad Height** cells. Right-click and select **Properties**.

34. Enter 80 (mils) for both **Pad Width** and **Pad Height**. Click **O.K**.

35. Next disable all layers for routing except the **BOTTOM**. Select **View Spreadsheet** in the toolbar. Select **Layers**.

36. In the **Layers** spreadsheet, click on the **Layer Type** cell to highlight the entire column. Press the **Ctrl** (on the keyboard) and click the **BOTTOM**.

37. Right-click and select **Properties**. In the **Edit Layer** dialog box, select the **Unused Routing** option.

38. Now all tracks will be routed on the bottom layer. This will simplify component soldering once the board is fabricated.

39. Select **Auto-> Autoroute -> Board**. The software will route the board.

40. When it is done, a pop-up window will announce that the task has been completed. Click **O.K**.

41. Ensure that all connections have been routed. Select a track and all connected tracks will be highlighted.

> **Hint: Unroute the board (select **Auto -> Unroute -> Board**) and move components around. Then re-try autorouting the board, or manually route the tracks.*

42. To manually create tracks, select **Tool** from the toolbar menu select **Track-> Tool**.

43. Use the mouse to route the tracks in the design.

44. In case you need to edit the widths of any tracks created by the autoroute, select **Tool** from the toolbar menu select **Track-> Tool**. (If not, proceed to Step 54).

45. Press the **Shift** key, place the crosshair on a track and click the left mouse button.

46. Right-click the mouse to open the pop-up window. Select **Change Width**.

47. In **Track Width** box, enter the new width, 50 (mils). And mark the Net box (this changes all the tracks connected to the same node). Click **O.K**. Repeat until all the tracks have been modified.

<u>Create board outline</u>

48. Click on the **Obstacle Tool**  in the toolbar.

49. Place the crosshair of cursor on the black field of the design window and double-click. The **Edit Obstacle** window should appear.

50. Select **Board Outline** for the **Obstacle Type.**

51. Change the **Width** to **62 mils**.

52. Select **Global Layer** for the **Obstacle Layer**. Click **OK**.

53. Begin drawing enclosed border of the design.

54. Proceed to **Creating Files**.